Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 127 (2015) 809 – 815

Procedia Engineering

www.elsevier.com/locate/procedia

International Conference on Computational Heat and Mass Transfer-2015

Numerical Study of Flow Structure Developing Around Double Suction Pipe of a Pumping System

M. S. Tawade[a*], R. S. Maurya[b]

[a] A Post Graduate Student, Department of Mechanical Engineering, Sardar Patel College of Engineering, Mumbai, India
[b]Department of Mechanical Engineering, Sardar Patel College of Engineering, Mumbai, India

Abstract

The vortex formation at pump intake is a common problem encountered by practicing engineers of pumps. The vortex formation due to low submergence and high intake velocity leads to mechanical damage to impeller and loss of pump performance. In order to meet the flow requirement, sometimes multiple pumping systems are used where suction pipes are located nearby. Because of this arrangement, the performance of the pumps gets influenced. The objective of the present work is to investigate the mechanism of flow structure developing around a suction pipe of a pump. Work has been executed using ANSYS Fluent. The flow patterns of double pipe pump intake system is examined and discussed under various flow conditions. The suction effect on approaching flow near the bell mouth is examined using numerical modeling. The swirl angle in pump intake system is used to evaluate its performance. The results indicate that the pump becomes less efficient with decrease in submergence depth.

Peer-review under responsibility of the organizing committee of ICCHMT – 2015

Keywords: Vortex; Suction; Sump; VOF; Fluent; Multiphase.

1. Introduction

In the pumping system role of suction line and the sump are significant for its uninterrupted and satisfactory operation. Sewerage pumping system, power generation systems, cooling systems and processing system are some key applications where compromise in pump's operation cannot be tolerated. Pump's operation is often troubled by

*Corresponding author
*E-mail address:*mstawade@gmail.com

Peer-review under responsibility of the organizing committee of ICCHMT – 2015
doi:10.1016/j.proeng.2015.11.416

low efficiency which is caused by vortices, spiral flows and large-scale reverse flows evolving on the suction side of pump intake system. Deborah Isabel Bauer and Tatsuaki Nakato [1] experimentally studied flow field formed in multiple pump sump. The operating condition was varied to observe its impact on the formation of vortices around suction intake. In a numerical investigation, Constantinescu and Patel [2] used k-ε turbulence model to capture the flow pattern of pump intakes of a power plants. They were able to conclude that numerical model can predict the location, size and strength of vortices sufficiently. It was proved that the k-ε turbulence model is a robust model to capture turbulence generating on suction side of a pump, but it requires a rigorous validation. Rajendran et al. [3] modelled a simple pump sump using tools of CFD and validated their model with help of experimental data acquired through PIV (Particle Image Velocimetry) measurements of a physical model. The number vortices formed, their locations, and general structures predicted by numerical technique were found to be in good agreement with those observed in experimentation. However, the vortices were found to be generally larger and weaker than what predicted by the model. Okamura et al. [4] carried out a benchmark test on several CFD codes and validated using a small scale model. They conclude that some of the CFD codes predict the visible vortex location and occurrence accurately enough to be used in industry, although the results of magnitudes near the bell mouth are poor and only agree qualitatively. Li Hai Fang [5] investigates the formation and evolution of the free surface vortex by experimental model. It is found that the tangential velocity distribution is similar to that of the Rankine vortex and the radial velocity changes little in the vortex functional scope. Vortex starts from the free surface and gradually intensifies to air entrainment vortex. Zhan et al. [6] develop 3-D numerical model for pump intake based on the Navier-Stokes equations with the RNG k -ε turbulence model and the VOF method to simulate the free surface. Five different types of pump intake systems were investigated and it was concluded that pump performance gets affected by flow patterns due to vortices formed near the corners of inlet channel and intake bay. Kim et al. [7] studied flow uniformity according to the flow distribution in the pump intake channel to find out the cause of vortex occurrence by experiment and CFD. It was concluded that submerged vortex around the pump intake occurs on the bottom, side and rear walls. In order to obtain better use of water resource and decrease the risk related to loss of prime, Adrian [8] presented a method for mitigating the negative effect of vortex motion inside the suction chambers of centrifugal pumps. He studied the influence of a rotating device on the vortex motion of a vertical suction pipe. Wei-Liang et al.[9] analyzed pump sump flows with three dimensional large eddy simulation models and used an acoustic doppler velocimeter in his experimentation. Results indicate that the pump becomes less efficient with increasing discharge. Luca et al. [10] studied gas entrainment phenomenon occurring in nuclear plants due to the possibility of free vortex formation at the surface.

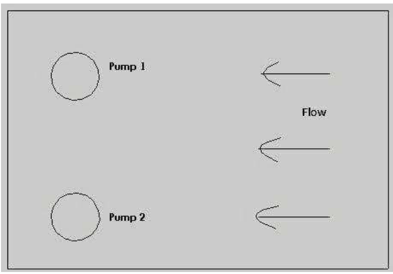

Fig.1. Double Pipe Pump Intake System

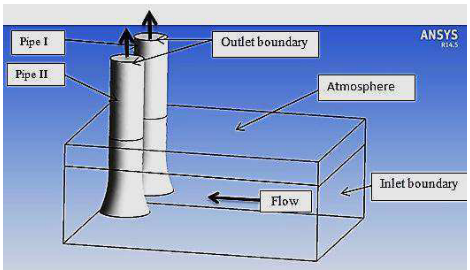

Fig.2. Computational Domain

The summery of literature shows that interfacial distortion occurring closer to pump suction at free surface has not been sufficiently addressed by the researchers. The depth of submergence of multiple suction pipe intake is expected to be crucial and worth of investigation to help pumping system designer to ensure pumping system's satisfactory performance. Present work tries to addresses these issues with the help of numerical computation. The problem under investigation consists of a sump receiving water from one end and being sucked by two suction line located nearby as shown in fig.1 and fig.2.The operating conditions of pump are being varied under different submergence depth. The objective of present work is to investigate the flow structure developing around double suction pipe and to know the effect of depth of submergence on the pump performance.

2. Governing Equation

The problem under investigation is a two phase flow of interfacial interaction. The flow is anticipated to be transient, turbulent and incompressible nature. The forces ruling the flow are shear, pressure gradient, surface tension and gravity. Mathematical model listed below represents behavior and characteristics of the system. As the problem is a two phase flow, the time dependent development of flow structure is captured through VOF model.

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \vec{v}\right) = 0 \tag{1}$$

Momentum Balance Equation:

$$\rho\left(\frac{\partial \vec{v}}{\partial t} + \vec{v}.\nabla\vec{v}\right) = -\nabla p + \nabla.\vec{\tau} + \rho\vec{g} + F \tag{2}$$

Volume Fraction Equation:

$$\frac{\partial\left(\alpha_q . \rho_q\right)}{\partial t} + \nabla.\left(\alpha_q . \rho_q \overrightarrow{v_q}\right) = \sum_{p=1}^{n}\left(m_{pq} - m_{qp}\right) \tag{3}$$

$$\sum_{q=1}^{n} \alpha_q = 1 \tag{4}$$

The velocity vector is $\vec{v}$= (u, v, w), P is the pressure, μ is the viscosity of the fluid, ρisthe density of the fluid, F is surface tension force and t is time. The tracking of the interface between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. For the q^{th}phase, equation (3) is used where m_{pq} is the mass transfer from phase p to phase q and m_{qp} is the mass transfer from phase q to phase p. The volume fraction equation will not be solved for the primary phase; the primary-phase volume fraction will be computed based on the constraint given by equation (4).

Boundary conditions are shown in fig.2.

Inlet: uniform velocity
Outlet: uniform velocity
Interface: atmospheric pressure
Walls: no-slip conditions.

3. Numerical Implementation

The problem is investigated using commercial software Ansys Fluent 14.5. In order to capture interfacial interaction compressive Volume of Fluid (VOF) model is used. The Pressure Implicit with Splitting of Operators (PISO) algorithm is used for the pressure-velocity coupling. The turbulence effect is captured by using realizable k-έ turbulence model. The Pressure Staggering Option (PRESTO) discretization scheme is used for pressure interpolation. To handle the convective effect, second order upwind scheme is used for momentum and turbulence equations. The bounded second order implicit scheme is used to get transient effect.

3.1. Model validation

To ensure reliability of selected numerical scheme for the proposed problem, a validation is carried out by simulating an experimental investigation done by Rajendran et al. (1999). The experimental setup consists of an intake pipe withdrawing water from rectangular channel. Quantitative comparisons of results pertaining to the vortices formed are shown in fig. 3.The vortex axes are considered at point of maximum vorticity. The tangential velocities with radial distance from this point are plotted. Numerical results accurately predict location and general structure of vortices. The results obtained from numerical model are in good agreement with those observed in experiment.

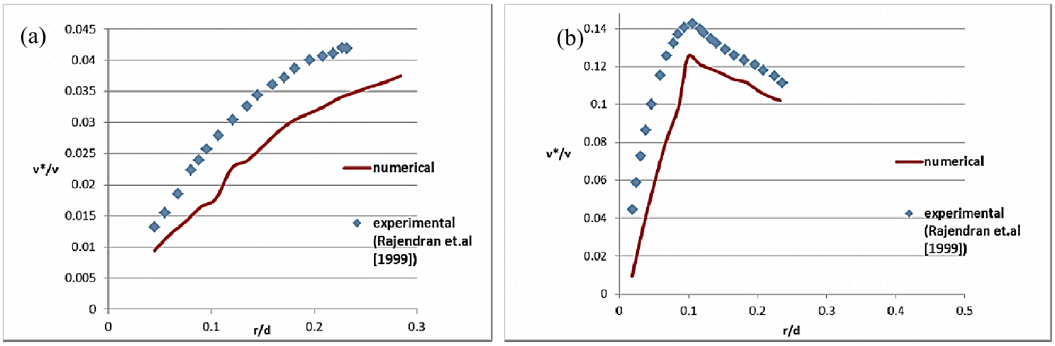

Fig.3. Comparison of Experimental and Numerical Results (a) Free Surface Vortex (b) Floor Vortex

4. Results and Discussion

The problem is investigated by considering two suction pipes of equal diameter with their bell mouth entry diameter D. Other relative geometrical dimensions of the simulation cases are stated below.

Distance of suction pipes from sidewall and back wall: 1D

Centre to centre distance between suction pipes: 2D

Floor clearance: 0.5D

Under suction flow condition of Re=330505, three cases of submergence depth, S = 1.5D, D and 0.75D, are considered to investigate the flow structure evolving in sump and their effect on pump performance.

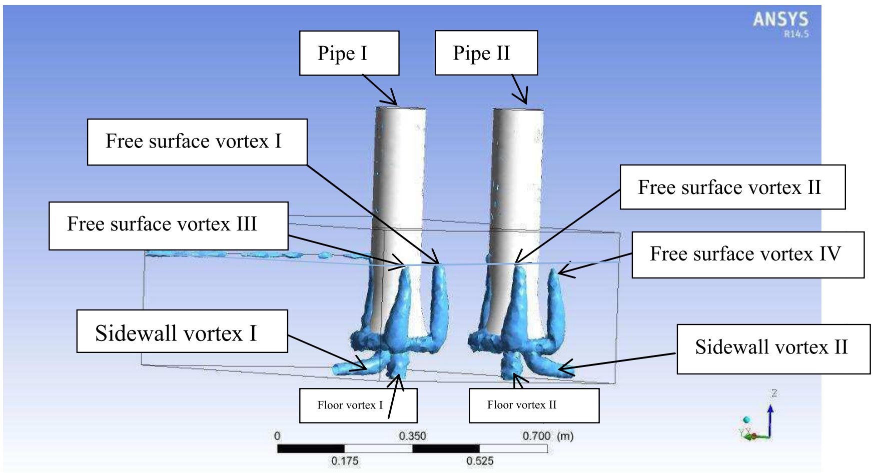

Fig.4. Main Vortices in Pump Sump

The presence of three distinct vortices - sub-surface vortices attached to the floor, side and surface vortices evolving in flow structure can be seen. They are found to exist simultaneously as shown in fig.4. The formation of vortex can be attributed to the formation high pressure gradient developing near bell mouth due to strong shear effect. A high value of eddy viscosity is expected in flow near the pump bell. With reduction in the submergence level in pump sump, a high shear stress zone is created near the bell mouth of suction pipes. The presence of high shear increases the vortices strength and propagates vortex formation in the locally favourable zones. The effect of depth of submergence on vortices is discussed below.

4.1. Effect on vortex location

The simulated result shows the presence of three vortices in all depth of submergence under consideration. The flow structure and topology is observed to be remaining same as shown in fig.5. It can be interpreted that forces (radial and tangential components) which are responsible for vortex formation are independent of depth of submergence. This can be attributed to the strength of pressure gradient which is created by imposed boundary condition remain same in all the cases. Two floor vortices formed below the pipes spinning in opposite direction are due to symmetric pump intake geometry, do not change their location. There are free surface vortices on either side of pipe, situated slightly behind the pipes. The vorticity plot [Fig.5 (d)] shows that well defined free surface vortex core are formed.

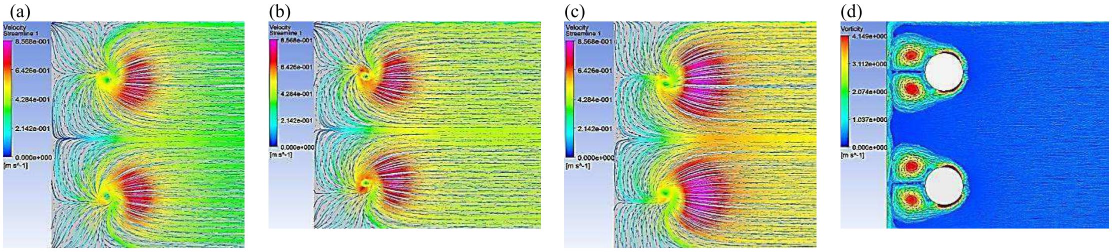

Fig.5. Streamlines of Floor Vortex in (a) S =1.5D (b) S =1D (c) S = 0.75D (d) Vorticity of Free Surface Vortex

4.2. Effect on vortex size

Table 1 shows quantitative evaluation of vortex size under different submergence depth compared to their size at submergence depth of 1.5D. The vortex axis located at point of maximum vorticity and position of maximum tangential velocity is used to calculate vortex size [Rajendran et al. (1999)]. In case of floor vortex I, size of vortex increase by 12% and 18% as submergence decreases from 1.5D to 1D; and from 1.5D to 0.75D respectively. It can be seen that size of free surface vortices I increase by 9% and 13.5% as submergence decreases from 1.5D to 1D; and from 1.5D to 0.75D respectively. The sidewall vortices also grow in size with decrease in submergence level as presented in Table.1.

Table.1. Percentage increase in the vortex sizes compared to the size of vortices at S =1.5D

Depth of Submergence S	Free Surface Vortex		Floor Vortex		Sidewall cortex	
	I	II	I	II	I	II
1D	9%	8%	12%	10%	10%	12.5%
0.75D	13.5%	15%	18%	17%	20%	22.5%

4.3. Effect on vortex strength

The vortex strength is measured by the parameter - flow circulation. The fig.6 shows variation of circulation with radial distance from the centre of the vortex. It is linear for all three vortices with different rate of variation. The circulation in free surface vortex I is found to increases by 67% as submergence decreases from 1.5D to 1D and 167% as submergence decrease from 1.5D to 0.75D. Similar pattern can be observed in floor and sidewall vortices also.

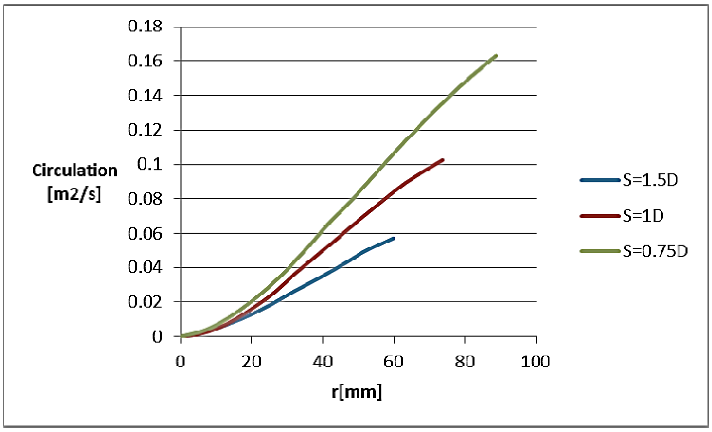

Fig.6. Effect of Submergence on Circulation in Free Surface Vortex I

4.4. Swirl angle

The vortex formation in pump sump increases the swirl in suction pipe. The swirl in suction pipe increases the turbulence in suction pipe and hence pump has to supply more power to lift the fluid.

The swirl angle θ is defined as

$$\theta = \tan^{-1} \frac{v_t}{v_a} \qquad (5)$$

Where v_t is the tangential velocity, v_a is the mean axial velocity in pipe. The Table.2 shows the swirl angle in pipe1 and pipe2 at various submergence depths. Positive value of swirl angle in pipe1 represents clockwise rotation of fluid whereas negative value of swirl angle in pipe2 represents fluid rotates in anticlockwise direction. It observed that swirl angle increases as submergence depth decrease. For submergence depth 1.5D, magnitude of swirl angle in pipe1 and pipe2 is 1.18 and 1.146 respectively whereas for submergence depth 0.75D, swirl angle magnitude for pipe1 and pipe2 is 5.02 and 5.14 respectively.

Table.2. Effect of submergence on swirl angles in pipes

Submergence	Swirl angle Pipe1	Swirl angle Pipe 2
1.5D	1.189	-1.146
1D	3.8162	-3.765
0.75D	5.02	-5.140

It indicates that the flow approaching the impeller has a rotational flow field that may oppose or add to the impeller rotation, depending on direction.

5. Conclusion

Present work deals studies of flow structure developing around a double suction pipe of a pumping system using numerical technique. Study is limited to the assessment flow structure evolving in the sump under varying depth of submergence and its impact on pump performance. The result authenticates the presence of different vortices near and around bell mouth entry. It show that the intensity of the free surface and subsurface vortices increases as the submergence is lowered. The swirl angle being an effective tool for evaluating pump performance and it is capable to show the impact of change in submergence level under given operating condition of a pump. Swirl angle value increases with decrease in submergence level. High value of swirl angle increases the turbulence in suction pipe. Hence Swirl angle value should be maintained as low as possible.

References

[1] I. Deborah, N. Tatsuaki N, Subsurface vortex suppression in water intake with multiple pump sumps. IIHR technical reportno.389, 1997.

[2] G.S. Constantinu, V. Patel, Numerical model for simulation of pump-intake flow and vortices. Journal of Hydraulic Engineering, 124(2) (1998) 123-134.

[3] V.P. Rajendran, S.G. Constantinu, V.C. Patel, Experimental validation of numerical model of flow in pump intake bay. Journal of Hydraulic Engineering, 125(11) (1999) 1119-1125

[4] T. Okamura, K. Kamemoto, J. Matsui, CFD Prediction and model experiment on suction vortices in pump sump, Proc. of 9th Asian International Conference on Fluid Machinery, Korea, Oct 16-19, AICFM9-053, 1–10, 2007.

[5] H. Li, H. Chen, Experimental and numerical investigation of free surface vortex. Journal of Hydrodynamics, 20 (4) (2008) 485-491.

[6] J. Zhan, B. Wang, L. Yu, Numerical investigation of flow patterns in different pump intake systems, Journal of Hydrodynamics 24(6) (2012) 873-882.

[7] C.G. Kim, Y.D. Choi, J.W. Choi, Y.H. Lee, A study on the effectiveness of an anti-vortex device in the sump model by experiment and CFD, IOP conference series: Earth and Environmental Science, 15(7) (2012) 072004.

[8] C. Adrian, Method for mitigating the negative effect of vortex motion inside the suction chambers of centrifugal pumps**.** Incas Bulletin*,* 5(3), 37-44.

[9] C. Wei-Liang, H. Shih-Chun, H. Kao-Shu, Numerical and experimental study of pump sump flows, Mathematical Problems in Engineering, 2014 (2014) 1-20.

[10] C. Luca, N. Matteo, C. Gianfranco, Experimental study on unstable free surface vortices and gas entrainment onset conditions, Experimental Thermal and Fluid Science 52 (2014) 221–229.